

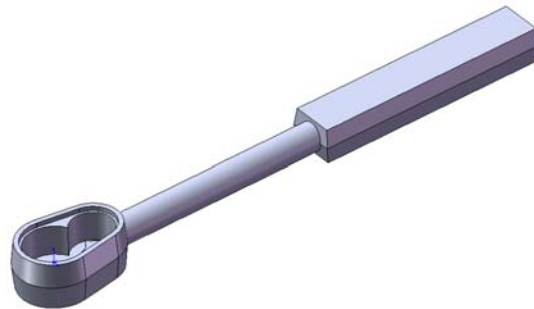
# SolidWorks Workshop

## Institute of Field roBOTics

### Advanced Parts


#### Exercise 1: Ratchet

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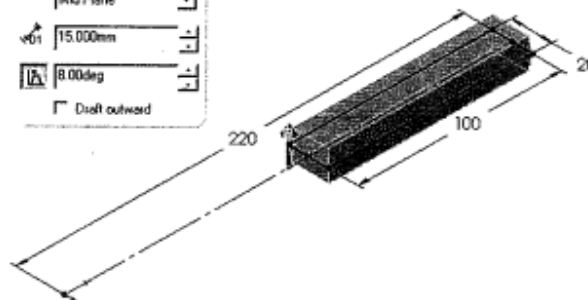
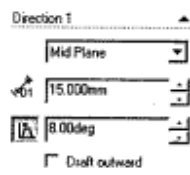


#### 9 Extrusion.

Choose the **Mid Plane** option from the list and enter a depth of **15mm**.

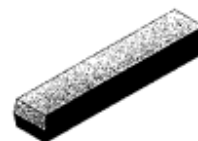
Click **Draft**  and set the angle to **8°**. The **Draft Outward** check box should be cleared.

Click **OK** to create the feature.

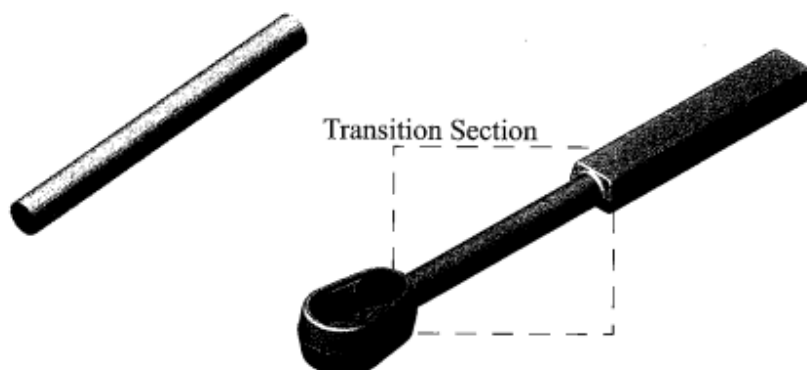


#### 10 Completed feature.

The completed feature is shown at the right. Name the feature **Handle**.

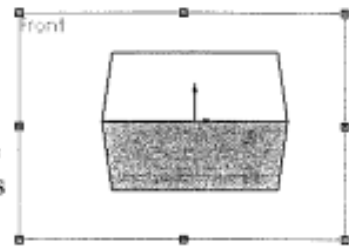


The second feature in the part is the **Transition**, another boss that will connect the **Head** to the **Handle** feature. The sketch for this feature is created on a standard reference plane.




#### 14 Normal To view orientation.

Using the **View Orientation** dialog box, change to the **Normal To** orientation. To do this, select the **Front** plane and double-click the **Normal To** option in the **View Orientation** dialog box. This orients the view so you can see the plane's true size and shape and makes sketching easier.



#### Tip

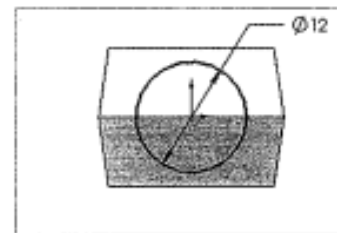
You can also select the plane and click the **Normal To** tool  on the Standard Views toolbar.

#### Sketching the Circle

Many inference points can be used to locate circles. You can use the center of previously created circles, the origin and other point locations to locate the circle's center. In this example, we will automatically capture a coincident relation to the origin by sketching the center of the circle on it.

#### 15 Add a circle and dimension it.

Using the **Dimension Tool**, add the diameter dimension to fully define the sketch. Set the value to be **12mm**. The sketch is fully defined.

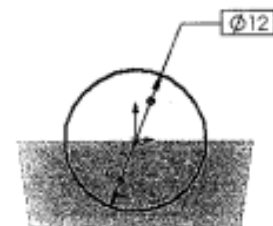


#### 16 Hide the Front reference plane.

With the dimensioning standard currently in use, diameter dimensions are displayed with the arrows inside the circle. You can change the display so that the arrows are outside of the circle.

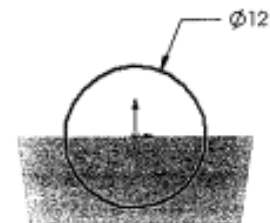
#### 17 Click the dimension.

Two small green dots will appear on the arrowheads of the dimension.



#### 18 Toggle the arrows.

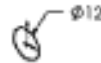
Click one of the green dots to toggle the arrows to the outside of the circle. This works on all dimensions, not just diameter dimensions.



#### Changing the Appearance of Dimensions

### 19 Change to Isometric view.

Unlike when you created the first feature, the system will not switch view orientations automatically for any other bosses or cuts. Use the **View Orientation** dialog box or the Standard Views toolbar to change to an Isometric view.

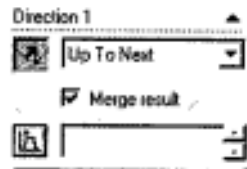


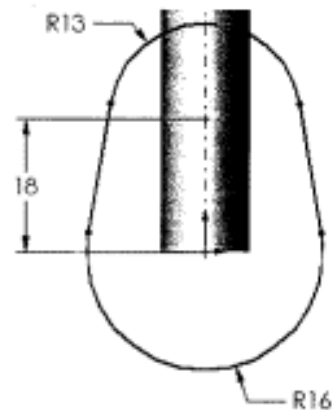
The sketch will be extruded up to the next face(s) it encounters along its path. It is important to watch the preview graphics to determine that the boss is going in the proper direction, reversing the direction if necessary.

### 20 Up To Next extrusion.

Click **Insert, Boss, Extrude...** and watch the preview display. Change the direction so that the preview shows the extrusion running towards the **Handle**. Change the end condition to **Up To Next**. Click **OK**.

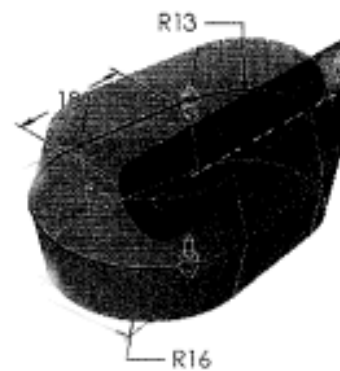
Rename the feature to **Transition**.





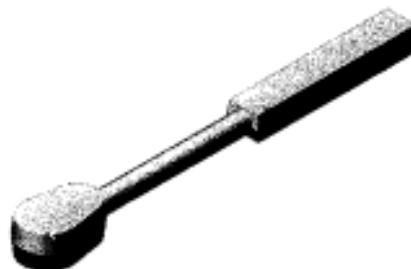
#### 41 The extrusion.

Change to an **Isometric** view and click **Insert, Boss, Extrude...** from the menu. Set the type to **Mid Plane**, depth to **20mm** and draft to **6°**.



#### 42 Completed Head.

The three main features that make up the overall shape of the part are now complete. Rename the latest feature to **Head**.



**49 Offset the face boundaries.**

With the face still selected click the **Offset** tool on the toolbar. Set the distance value to **2mm** and **Reverse** the direction if necessary, moving the offset to the inside.



**50 Resulting Offset.**

The offset creates two lines and two arcs. This geometry is dependent on the solid face it came from and will change with the solid. The sketch is automatically fully defined and ready to extrude as a cut.



**51 Settings for the cut.**

Choose a **Blind** cut with **2mm** for the depth value and click **OK**.



**52 Rename the feature.**

Change the name of the feature to **Recess**.

The **Pocket** is another cut feature, applied to a planar face of the model. This sketch uses overlapping circles that are trimmed to create a single contour. The centers of the circles are related to existing circular centerpoints.

**53 Open a sketch.**

Select the top, inner face created by the last feature as the sketch plane.

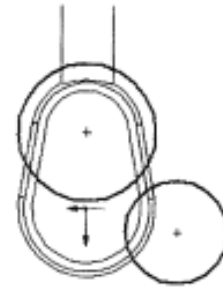


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#### 54 Sketch circles.

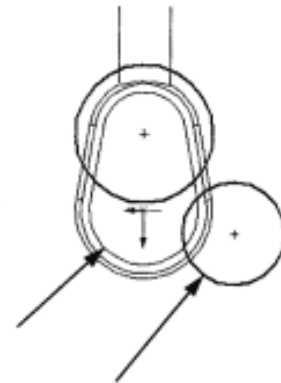
Using the **Circle** tool, create a circle using the existing centerpoint location as the circle's origin. Snapping to this location will relate the circle to it automatically. Create a second circle off to the side of the model.

Switch to the wireframe modes (**Hidden Lines Removed** or **Hidden Lines Visible**) only when necessary.



#### 55 Relate the centers.

Click **Add Relation**  to open the **Add Relations** PropertyManager. Select the second circle and the edge of the cut. Choose the **Concentric** option and click **OK**. **Concentric** forces the two arcs (the circle and the circular edge) to share a common center. This will pull the circle into position.




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### Trimming

Sketch entities can be trimmed shorter using the **Trim** option. In this example, the overlapping portions of the circles will be removed.

#### Introducing: Trim


**Trim** can be used to shorten sketch geometry. To lengthen, use the **Extend** command .

#### Where to Find It

- From the **Tools** menu, select **Sketch Tools, Trim**.
- Or, from the Sketch Tools toolbar pick the  tool.

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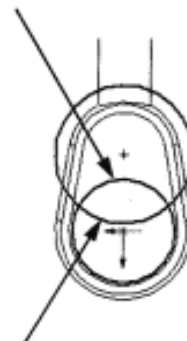
#### 56 Trim the circles.

Click on the **Trim** tool. .

#### Rule

*Pick the circles on the portions that you want to remove.*

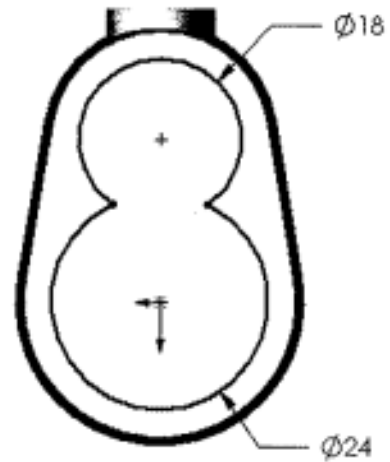
The system will find the intersections between the circles and remove the excess.



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**61 Results.**

The sketch is now ready to be extruded as a cut.



**62 Create the cut.**

Cut and extrude the sketch **13mm** down into the solid. Rename the feature **Pocket**.



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**63 Open a sketch.**

Click on the inner bottom “figure eight” face and open a new sketch.



**64 Create a circular hole.**

Sketch a circle centered on the upper center mark and add the dimension. Set the diameter to **9mm** and create a **Through All** cut.

Name the feature **Wheel Hole**.



## Exercise 2: Pulley

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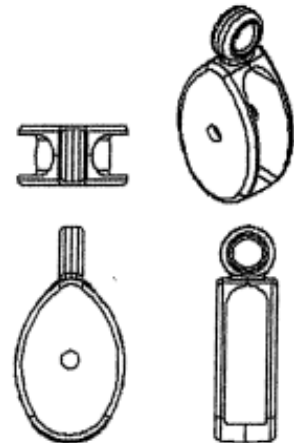
This Exercise reinforces the following skills:

- Creating draft while extruding
- Mid plane extrusion
- Filleting
- Symmetry

### Design Intent

Some aspects of the design intent for this part are:

1. Part is symmetrical.
2. All fillets are **1mm** unless noted.
3. Draft is **6°** on both body and hanger.



### Procedure:

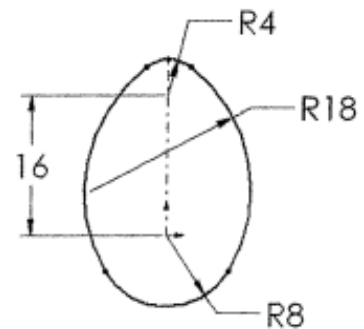
Open a new part using the Part\_MM template.

#### 1 Base feature.

Create the sketch of the base feature on the Front reference plane.

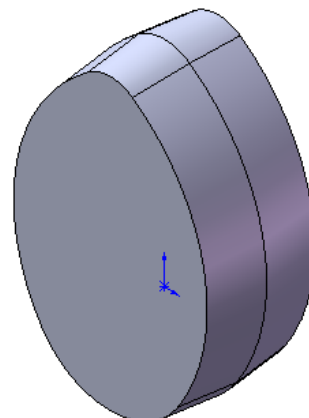
#### Tip

Go to the Index of the SolidWorks 2001 Help Topics. Look up **Arcs** and the options **Tangent** and **Centerpoint**.



#### 2 Extrusion with draft.

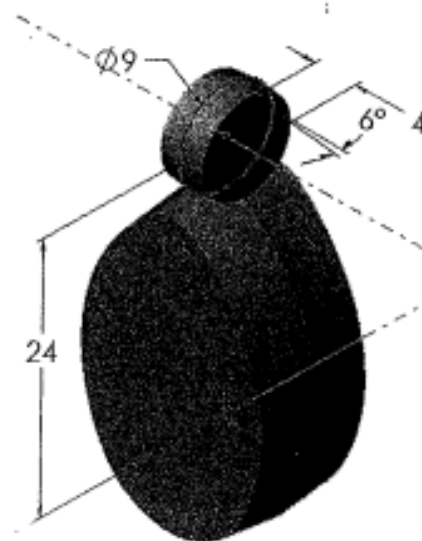
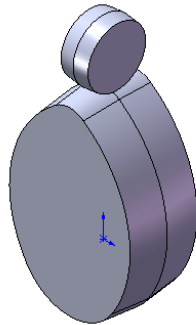
Extrude the sketch **10mm** using the **MidPlane** end condition and **6°** of draft.





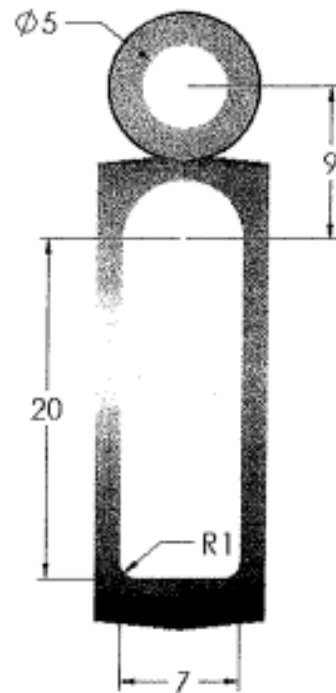
### 3 Hanger.

Create the hanger using a sketch dimensioned to the origin and another **MidPlane** extrusion with the same amount of draft.

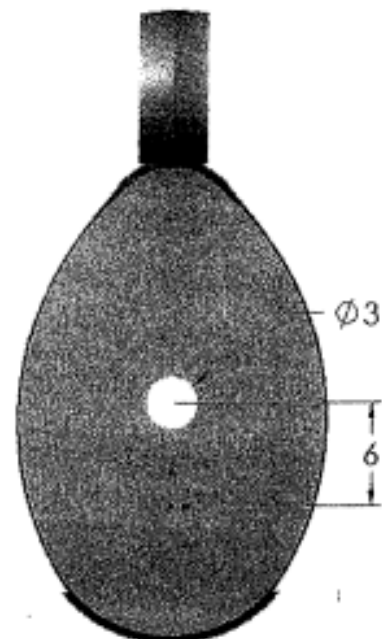


### 4 Cuts and fillets.

Create two through all cuts as separate features. One cut is circular through the hanger, the other through the main body. Add the fillet (1mm) to the edges after the cut.



Create a third **Through All** cut centered above the origin.



**5 Fillets.**

Add fillets of **0.5mm** and **1mm** as shown. Note that these fillets are very order dependent; the **1mm** fillets must precede the **0.5mm** ones.

**6 Save and close the part.**

